EFFECT OF OBSTACLE-SIZE ON THE FLOW STRUCTURE OF TWO-PHASE FLOW THROUGH ENLARGING CHANNEL

A thesis submitted in partial fulfillment of the requirements for the degree

of

Bachelor of Technology

in

Mechanical Engineering

by

Sabyasachi Mohanty (Roll No. 111ME0219)

under the guidance of

Dr. Suman Ghosh

DEPARTMENT OF MECHANICAL ENGINEERING
NATIONAL INSTITUTE OF TECHNOLOGY ROURKELA
JUNE, 2015

CERTIFICATE

*This is to certify that the thesis entitled "**Effect of Obstacle-Size on the Flow Structure of Two-Phase Flow through Enlarging Channel**", submitted by **Sabyasachi Mohanty (Roll Number: 111ME0219)** to National Institute of Technology, Rourkela, is a record of bona fide research work under my supervision, to the best of my knowledge in partial fulfilment of the requirements for the degree of **Bachelor of Technology** in the Department of Mechanical Engineering, National Institute of Technology Rourkela.*

Place: Rourkela
Date:

Dr. Suman Ghosh
Assistant Professor
Department of Mechanical Engineering
National Institute of Technology Rourkela
Rourkela-769008, Odisha, India

DECLARATION

I certify that

a) The work contained in the thesis is original and has been done by myself under the general supervision of my supervisor(s).

b) The work has not been submitted to any other Institute for any degree or diploma.

c) I have followed the guidelines provided by the Institute in writing the thesis.

d) I have conformed to the norms and guidelines given in the Ethical Code of Conduct of the Institute.

e) Whenever I have used materials (data, theoretical analysis, and text) from other sources, I have given due credit to them by citing them in the text of the thesis and giving their details in the references.

f) Whenever I have quoted written materials from other sources, I have put them under quotation marks and given due credit to the sources by citing them and giving required details in the references.

Date: Signature of the Student

Place: Sabyasachi Mohanty

ACKNOWLEDGMENT

I take this opportunity to express my deep gratitude to my Supervisor Dr. Suman Ghosh for his support during this project. I am greatly thankful to him for giving me this opportunity to work with him and for having faith in me throughout the course of completion of this project. His valuable advice and persistent encouragement were pivotal in completion of this project.

I also like to thanks to the staff of Mechanical Department for providing me necessary amenities to finish this project.

Last but not the least; I would like to thank my friends for their constant motivation without which this project would not have been successful.

Place: SABYASACHI MOHANTY

Date : Roll Number: 111ME0219

CONTENTS

LIST OF FIGURES

LIST OF EQUATIONS

LIST OF SYMBOLS

Symbol	Description	Unit
$F^{\rightarrow}$	Body Force	(N)
$g^{\rightarrow}$	Acceleration due to gravity	(m/s^2)
n	Number of phases	(-)
t	Time	(s)
α_k	Volume fraction of phase k	(-)
ρ_m	Mixture density	(kg/m^3)
$\vec{v_m}$	Mass averaged velocity	(m/s)
$\vec{v_{d}}_{r,k}$	Drift velocity of secondary phase k	(m/s)
μ_m	Viscosity of mixture	(N/m.s^2)

SUBSCRIPTS

Symbol	Description
a	Air
w	Water
m	Mixture
k	Secondary phase

ABSTRACT

A numerical investigation is made to study the effect of obstacle on the flow structure of air-water two-phase flow through a rectangular enlarging channel. The cross-section of the obstacle is considered as circular in shape. Effect of the diameter of the obstacle on the flow structure, pressure fluctuation and pressure distribution is made. Finite volume method (FVM) with mixture model and Eulerian model have been adopted for the present numerical simulation. Simulation results are obtained for both steady and unsteady cases. Due to change in both magnitude and direction of local velocities of fluid as the mixture approaches the obstacle, there is a marked difference in phase distribution around the obstacle. As the size of the obstacle increases, this becomes more remarkable. It can also be noticed that vortices before and after the obstacle becomes stronger with increase in size of the obstacle. The pressure difference is also monitored. It is found that as the obstacle size increases, the pressure drop increases and pressure distribution changes drastically.

Keywords: Air-water two-phase flow, Rectangular enlarging channel, Flow structure, Pressure fluctuation, Pressure distribution, Finite Volume Method (FVM), Mixture model, Eulerian model, Steady state, Unsteady state.

CHAPTER 1

INTRODUCTION
AND
LITERATURE SURVEY

1.1 INTRODUCTION

In nature and engineering problems, flows encountered are generally consisting of a mixture of phases. Single phase flow rarely occurs in nature. Simply a phase can be stated as one of the three states of matter which can be liquid, solid or gas. However, when this term is applied to fluid flow problems, it is done in a much wider sense. In a flow consisting of different phases, a phase is visualized as any collection of material that can be identified and has a distinct interaction with and inertial response to the enveloping flow and potential field.

The most uncomplicated case of multiphase flow is a flow consisting two phases. It follows all the fundamental laws of fluid mechanics but are more numerous and far more complicated than a flow consisting only one phase. Flows consisting of a gas and liquid or vapor and liquid are very common in engineering applications and industrial systems. They find application in chemical and nuclear reactors, power generation units, petroleum extraction units, biomedical processing units, safety valves, heat exchangers etc. Therefore the study of two phase flow becomes imperative for better and safe design of these units.

The problem inherent in such studies is that the two phase flow is substantially sensitive to geometrical variations of the system and topology of the components in the flow. The flow pattern is also much varied, ranging from bubble, slug, plug, stratified, wavy to dispersed flow. There are many other complexities involved in the basic understanding of multiphase flow in general and two phase flow in particular. Addition of obstacles in the flow path further increases the complexity. As such considerable research is required in this field.

1.2 LITERATURE SURVEY

Attempts have made in time to time, to study the effect of obstacles on the two phase flow structures and characteristics. Eames et al. (2004) have estimated mean-velocities of the flow through groups of fixed or moving bodies in domains that are bounded or unbounded. The domains lie inside a defined perimeter. Important concepts of Eulerian spatial mean velocity, which is mainly related to momentum, in fluid volume between the bodies and for flow outside the group, was introduced. Lagrangian mean velocity for fluid particles passing through the group of bodies was also introduced. The "full cavitation model" in which the CFD results of cavitation in liquid nitrogen flow through hydrofoils and ogives were presented by Zhang et al. (2008). To evaluate the performance prediction from the cryogenic fluids point of view, the model was re-examined assuming thermal equilibrium between liquid and vapor phase. It was applicable for the prediction of cavitation in liquid nitrogen taking into considerations thermodynamic effects. Zhou et al. (2011) using lattice Boltzman method besides a non-equilibrium extrapolation method, studied the structure of vortexes and particle dispersion in flows past a circular cylinder. Lagrangian framework was used to trace the particles. Effect of Reynolds number on vortex evolution was observed. The drag and the lift coefficients along with Strouhal number were in good agreement with previous studies. Talimi et al. (2012) reassessed mathematical investigations on heat transfer and hydrodynamic features of flows consisting of two phases in small tubes and channels. The review was organized into two classes, circular and non-circular. Different facets such as formation of slug, its shape, pressure drop, flow pattern, heat transfer, etc. were considered. Haoran (1993) was involved in the study of distribution of phases in low quality dispersed flows consisting of two phases. The study comprised of theoretical as well as experimental parts. The theoretical part was more general while the experimental part focused on bubbly flows past obstacles in vertical tubes. The theoretical study was mostly concerned with improving modeling algorithms for estimating phase distribution around obstacles and with gaining a better picture of the interaction between dispersed phase and the fluid carrying it, as the dispersed phase moves past the obstacle. Emrah (2011) analyzed adiabatic air water flow through a horizontal channel having smooth expansion. Eulerian model was used for the analysis. The internal diameter of the channel was increased from 32 mm to 40 mm. Water flow rate was kept constant at 3l liters/sec. Air flow rate was taken as 50 and 61 liters/sec. Habeeb et al. (2013) performed experimental and numerical studies of the two phase flow phenomena around multi shape obstacles in an enlarging channel. Experiments were conducted for different air water flow rates. Experiments were done with an aim to visualize the flow structure as well as to get data for pressure drop which were generated using a pressure transducer. Numerical simulations were carried for both steady and unsteady cases. Results showed that when air or water discharge increases, turbulence increases and mean pressure difference increases. And circular obstacle records more pressure drop than square and triangular obstacles. Bon (2011) investigated two different air water interfacial flows including plunging wave breaking and flow past a vertical surface-piercing circular cylinder. The flow features near the air water interface showed significant changes with different Reynolds number from subcritical to critical regime.

1.3 GAPS IN THE LITERATURE

Recent researches on multi-phase flow through an enlarging channel having obstacle and are limited to the best of the author’s knowledge. More ever investigations using numerical techniques have not been done so much with this topic, though limited resources and time are required for it.

1.4 AIMS AND OBJECTIVES

Here, the objective is to capture the effect of obstacle size (by changing the diameter of a cylindrical obstacle) on the flow structure and pressure distribution of an air-water two phase flow through a rectangular enlarging channel using numerical simulation. Numerical investigations have been made on the flow behavior considering both steady and transient cases.

1.5 ORGANIZATION OF THE THESIS

This thesis has been organized into five chapters. Chapter two provides a detailed description of the problem investigated, while Chapter three deals with the details of solution methodology. Chapter four consists of results and discussions, where important results have been given followed by their discussion. Chapter five consists of important conclusions drawn and scope of future work.

CHAPTER 2

PROBLEM DESCRIPTION

This chapter is concerned with the detailed descriptions of the problem with schematic representation and corresponding physical boundary conditions.

2.1 PROBLEM DESCRIPTION

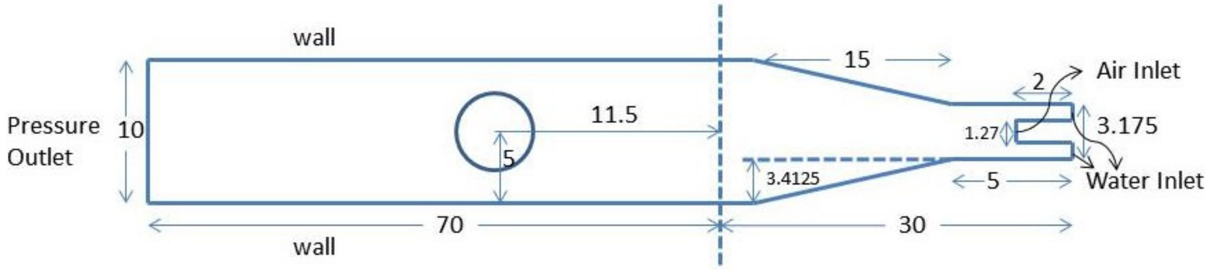


Figure 2.1 Schematic representation of the problem set up (all dimensions in cm).

The geometry of the problem has been taken from Esam and Riyadh, (2012). The test section is a rectangular channel having a cylindrical obstacle. The dimension of the rectangular cross section is 10 cm × 3 cm and the length of the test section is 70 cm. The diameter of the obstacle varies and its center is located at a distance of 115 cm from the test section entrance. There is an enlarging inlet section of length 30 cm before the test section. Therefore the total length of the setup is 100 cm. In the inlet section, there is a circular inlet port for air and an annular inlet port for water. The diameter of the air inlet port is 1.27 cm where the inner and outer diameter of the annular water inlet port are 1.27 cm and 3.175 cm respectively. The air and water are mixed in the enlarging portion before entering into the test section.

At the air inlet port, the velocity of air is specified as 1.5 m/s whereas at the water inlet port, the velocity of water is specified as 0.5 m/s. The outlet port of the test section is escaped to the atmosphere and pressure at outlet port is specified as 1 atm (101325 Pa). The walls are defined with no slip and no penetration criteria. Obstacles of different diameters are used for different cases. In case 1, the diameter of the obstacle is considered as 3 cm whereas case 2 and 3 consist of obstacle with diameter

4 cm and 5 cm respectively. Both steady and unsteady numerical simulations are considered for each of the above cases.

CHAPTER 3

NUMERICAL ANALYSIS

2D Eulerian and mixture models have been used to model the flow. Eulerian model is used for modeling multi-phases which are separate but interact with each other. The phases may be liquid, solid or gas. In this model, all the phases are under a same pressure. For every phase, solution is found out for momentum and continuity equations. A number of interphase drag coefficient functions are provided. All of the turbulence models are provided in an Eulerian model. Mixture model is a simplified model useful in modeling flows consisting of multiple phases, where the velocities of phases are different. However, they attain equilibrium locally over small spatial lengths. The phases should have a strong coupling. The model has the ability to model n number of phases. It is done by solving the equations of momentum, energy and continuity for the mixture. For secondary phases, equations of volume fraction are solved. In the present numerical model, water is taken as the primary phase and air is taken as the secondary phase. For relative velocities, algebraic equations are used. It is useful for bubbly flows, where the volume fraction of gas stays low.

For turbulent modeling, k-ε turbulent model has been used. It is one of the most used turbulence models. It has two additional transport equations for simulating the turbulent properties associated with the flow. The two transport variables k and ε denotes turbulent kinetic energy and turbulent dissipation rate respectively. Turbulent dissipation variable determines the extent of the turbulence, whereas k represents the amount of energy involved in turbulence.

3.1 MESHING

The software workbench has been used to create geometry and mesh. Grids have been generated using assembly meshing scheme. The model after meshing contained 28239 nodes and 27820 elements. Mesh adopted for the present problem, is shown in Figure 3.1

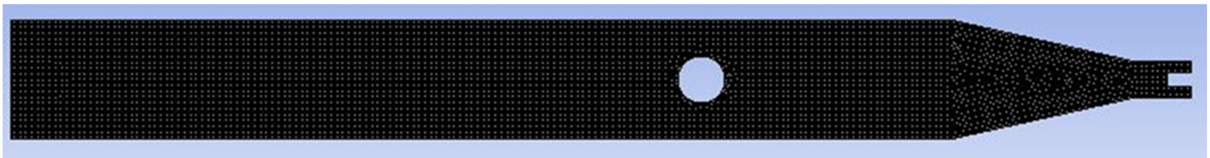


Figure 3.1: Mesh adopted for the present problem

3.2 GOVERNING EQUATIONS

The hydrodynamics of a flow, consisting of two phases, is captured by the mass and momentum conservation equations along with an added equation for advection to determine the interface of gas and liquid. As the axial drop of pressure is small, the flow can be considered as incompressible. The continuity equation:

$$\frac{\partial}{\partial t}(\rho_m) + \nabla.(\rho_m \vec{v_m}) = 0 \qquad (3.1)$$

Here $\vec{v_m}$ is the velocity averaged over mass and is given by: $\vec{v_m} = \frac{\sum_k^n \alpha_k \rho_k \vec{v_k}}{\rho_m}$. Here ρ_m is the density of mixture. Volume fraction of phase k is given by α_k and density of phase k is given by ρ_k. To obtain the momentum equation for the mixture as a whole, the individual momentum equation for each phase is to be summed over. It can be represented as:

$$\frac{\partial}{\partial t}(\rho_m \vec{v_m}) + \nabla.(\rho_m \vec{v_m} \vec{v_m})$$

$$= -\nabla p + \nabla.[\mu_m(\nabla \vec{v_m} + \nabla \vec{v_m}^T)] + \rho_m g^{\rightarrow} + F^{\rightarrow} + \nabla(\sum_{k=1}^n \alpha_k \rho_k \vec{v_{dr,k}} v^{\rightarrow}) \qquad (3.2)$$

Here the number of phases has been donated by n, body force by $F^{\rightarrow}$ and the mixture viscosity by μ_m. The mixture viscosity is defined as follows: $\mu_m = \sum_{k=1}^n \alpha_k \mu_k$. The drift velocity for secondary phase is given by $\vec{v_{dr,k}}$. It is defined as follows: $\vec{v_{dr,k}} = \vec{v_k} - \vec{v_m}$.

3.3 BOUNDARY CONDITIONS

3.3.1 Air inlet: At the air inlet port, the velocity of air is specified. $V_{a,in}$ = 1.5 m/s. At this air inlet port, volume fraction of air is considered as one. Velocity of the water is given as zero. Conditions for mixture is specified by giving turbulence intensity and hydraulic diameter.

3.3.2 Water inlet: At the water inlet port, the velocity of water is specified. $V_{w,in}$ = 0.5 m/s. At this water inlet port, volume fraction of air is considered as zero. Velocity of the air is given as zero. Conditions for mixture is specified by giving turbulence intensity and hydraulic diameter.

3.3.3 Pressure outlet: At the outlet port of the test section, pressure is specified. P_{out} = 101325 Pa. Conditions of mixture at the outlet is specified by giving turbulence intensity and hydraulic diameter.

3.3.4 At walls: Walls are defined giving no slip (V_x = 0 m/s) and no penetration boundary conditions (V_y = 0 m/s).

3.4 RESIDUALS AND CONVERGENCE

The residual convergence criteria for field variables, namely continuity, *x* velocity, *y* velocity, *k*, ε, are chosen as 10^{-6}. For unsteady cases, fixed time step is used. Time step size is used as 0.0001 seconds and maximum 40 number of iteration per step is considered.

3.5 VALUES OF THE PARAMETERS FOR SIMULATION

The turbulent kinetic energy (k) has been set as 0.0003375 m^2/s^2 whereas, the turbulent dissipation rate (ε) has been consider as 0.0007620108 m^2/s^3. Density of water and air have been given as 998.2 kg/m^3 and 1.225 kg/m^3 respectively. Viscosity of water has been taken as 10.03×10^{-04} kg/m.s and that of air has been taken as 1.7894×10^{-05} kg/m.s. Interfacial tension at air water interface has been given as 0.072 N/m.

CHAPTER 4

RESULTS AND DISCUSSION

Results of the problem obtained after solving the problem as defined in the chapter 2 are categorically provided here. At first results corresponding to grid independence test have been provided. Then the results obtained from the present methodology have been validated with the Esam and Riyadh, 2012 and have been provided. After validation, the results related to the present problem using the methodology (described in chapter 3) have been provided.

4.1 GRID INDEPENDENCE TEST

Grid independence test is performed to get the minimum number of grids which can give the correct results. Using less grids, the space or memory required to store the data will be less. Secondly it saves time which is more important as the solutions take large time to converge. Here to do the grid independence test, meshes having 27000, 28000 and 29000 nodes were generated and area weighted static pressure difference was calculated. The flow regimes corresponding to each of the cases, have also been shown. The velocity of air at inlet port is 1.32 m/s and that of water is 0.5 m/s at water inlet port. The radius of obstacle is 3 cm. As can be seen in Figures 4.1 and 4.2 there is not much difference in results for 28000 and 29000 nodes. Therefore result doesn't vary much after 28000 nodes and it can be used for subsequent simulations.

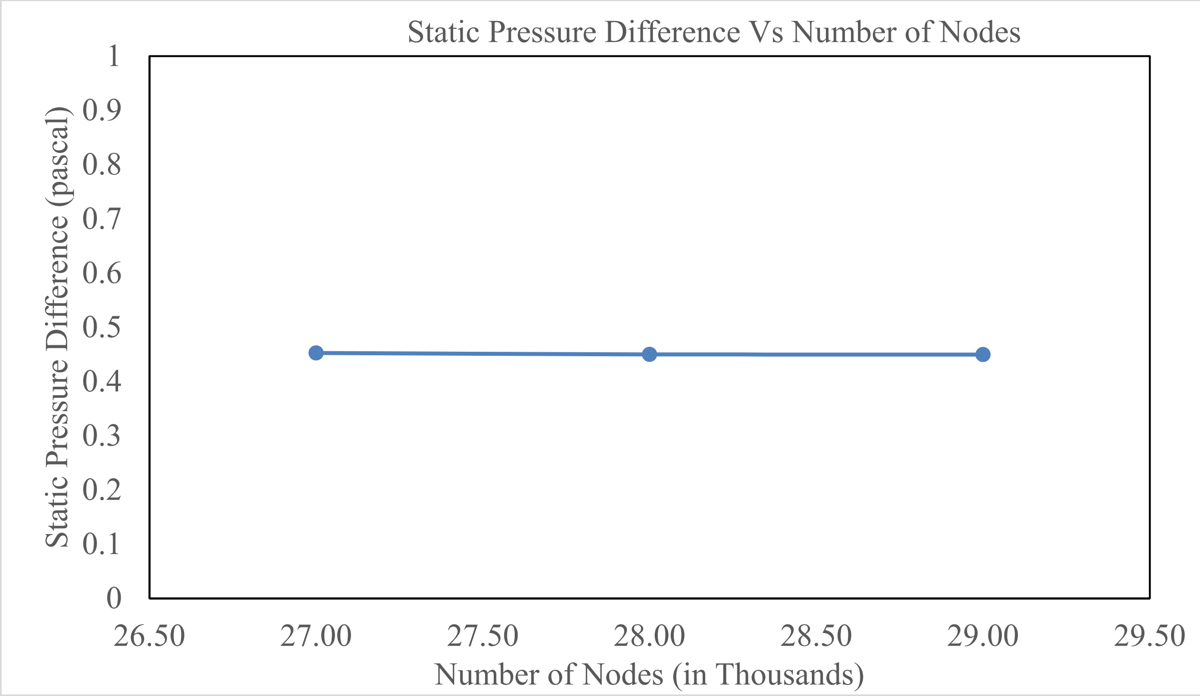
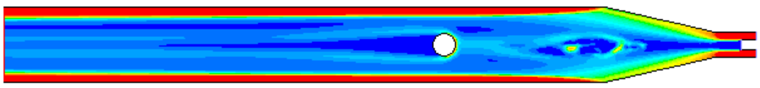
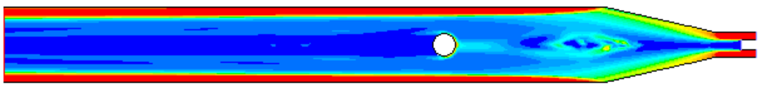
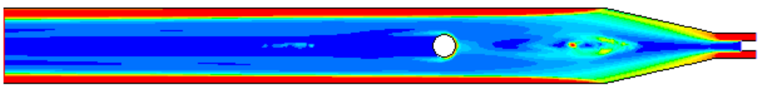


Figure 4.1: Static pressure difference versus number of nodes for water and air velocity of 0.5 m/s and 1.32 m/s respectively with radius of obstacle as 3 cm.

Number of nodes	Flow Structure
27000	
28000	
29000	

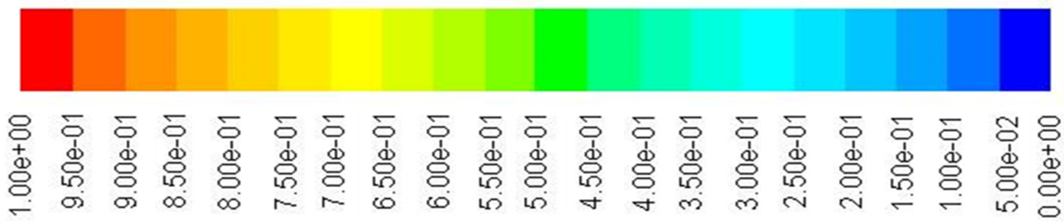


Figure 4.2: Flow structure using different number of nodes for water and air velocity 0.5 m/s and 1.32 m/s with radius of obstacle as 3 cm.

4.2 RESULT VALIDATION

In this section, validation of the present methodology has been done with that of an experimental work done by Esam and Riyadh, 2012. The flow structure for both the cases have been shown in Figure 4.3 where the inlet conditions are as V_w = 1.12 m/s and V_a= 2.63 m/s. The diameter of the obstacle is 3 cm. Both the cases in Figure 4.3, have been obtained using the same geometry and boundary conditions. As can be seen from figure 4.3, there are no such differences in the results obtained from the numerical analysis and that from Esam and Riyadh, 2012.

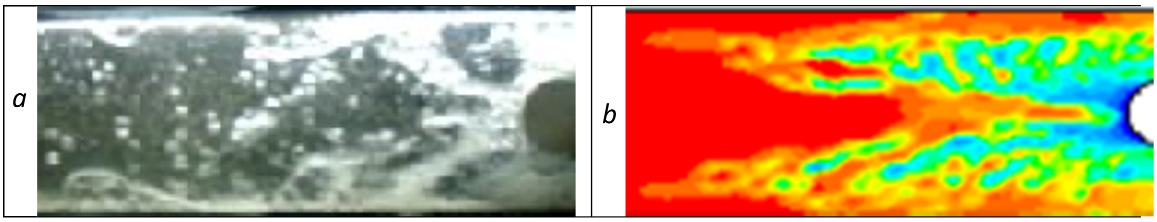


Figure 4.3 Comparison between the experimental result of Esam and Riyadh, 2012 and the present numerical results: (a) Through experimental (Esam and Riyadh, 2012), (b) Through present numerical simulation.

4.3 STEADY STATE RESULTS

4.3.1 Effect of Diameter of the Obstacle on Flow Structure: In the following pictures, steady state phase distributions for different diameters of the obstacle have been shown. It can be seen from Figure 4.4 that the flow structure becomes annular in nature after the obstacle.

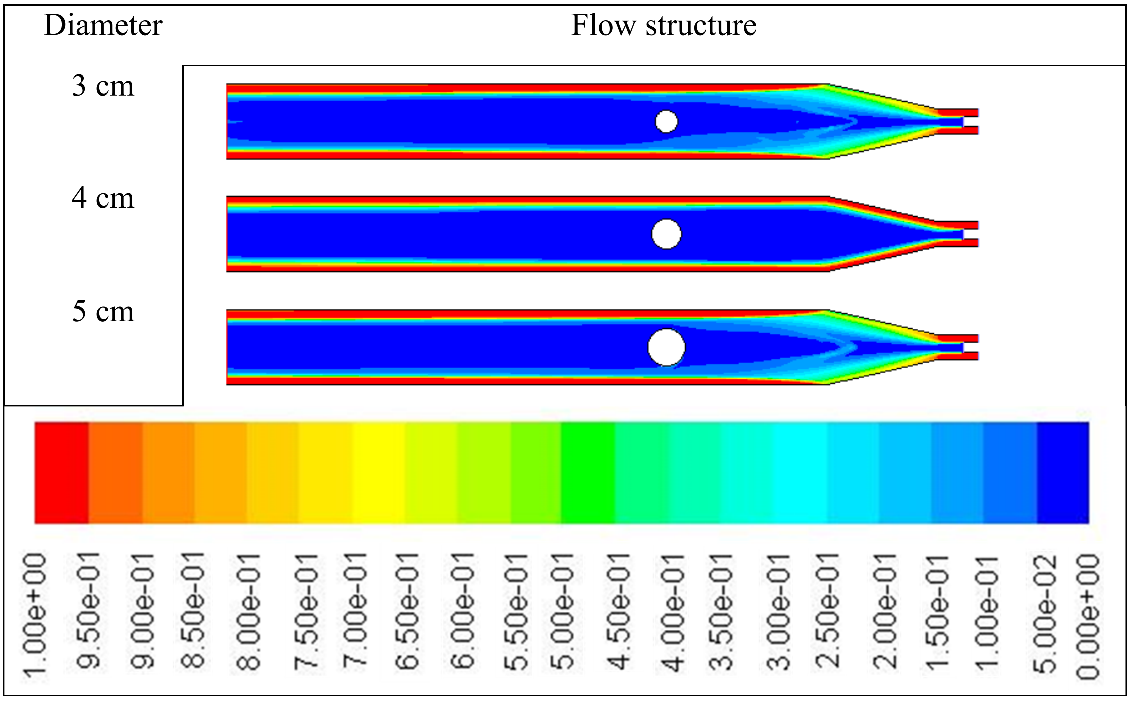


Figure 4.4: Flow structure for different diameter of obstacle (Steady state cases)

4.3.2 Effect of Diameter of the Obstacle on Static Pressure Distribution: This section deals with the variation of pressure along the axis for different diameter of the obstacle in steady state conditions. Figures 4.5 to 4.7 show this results.

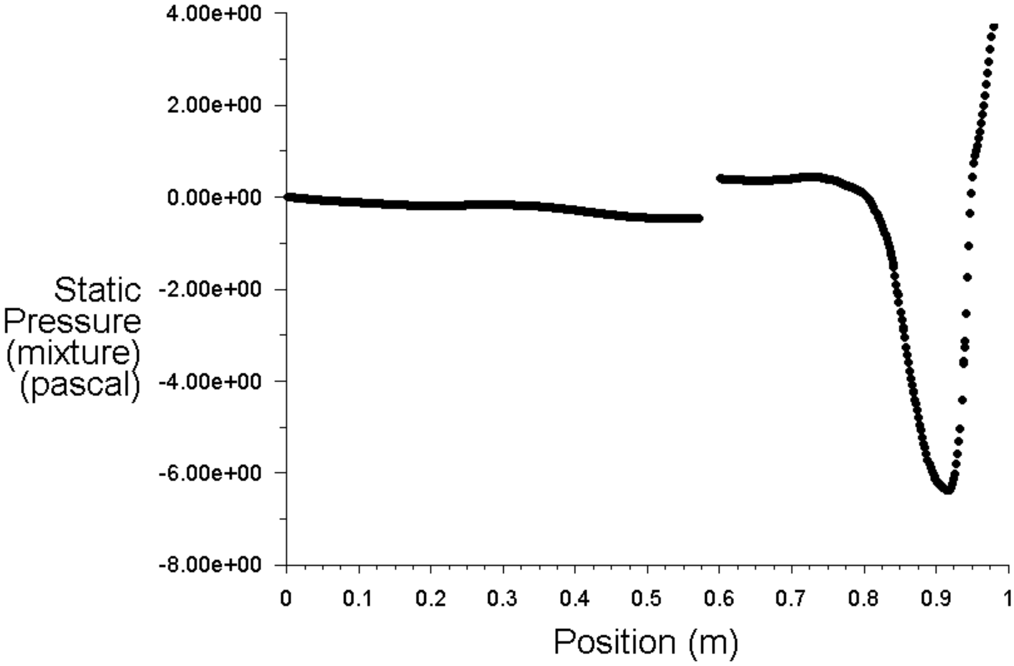


Figure 4.5: Variation of static pressure along the axis for the obstacle with diameter = 3 cm.

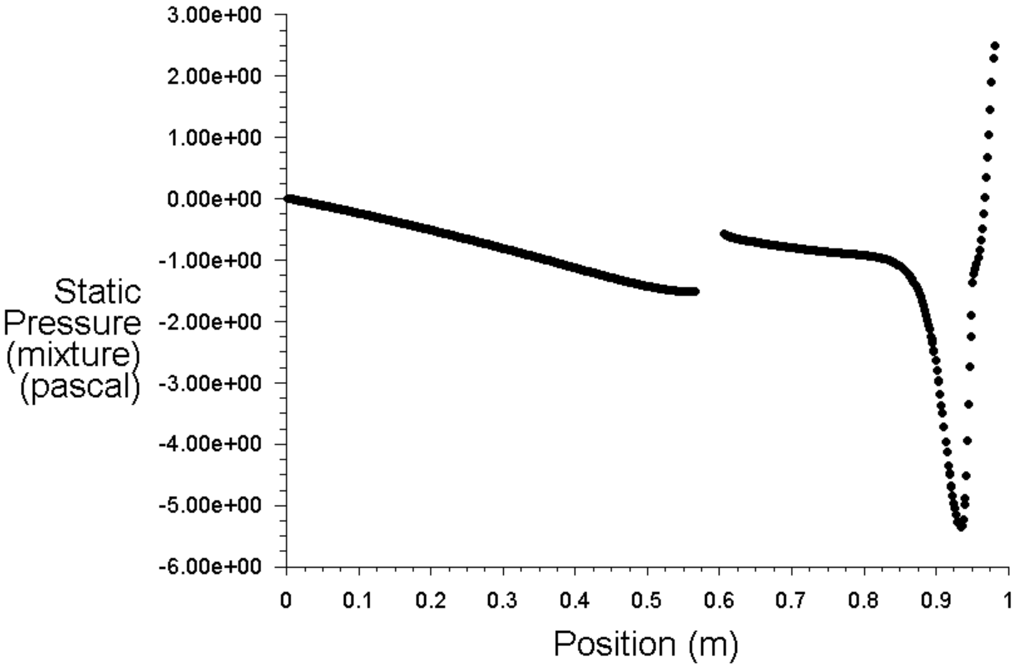


Figure 4.6: Variation of static pressure along the axis for the obstacle with diameter = 4 cm.

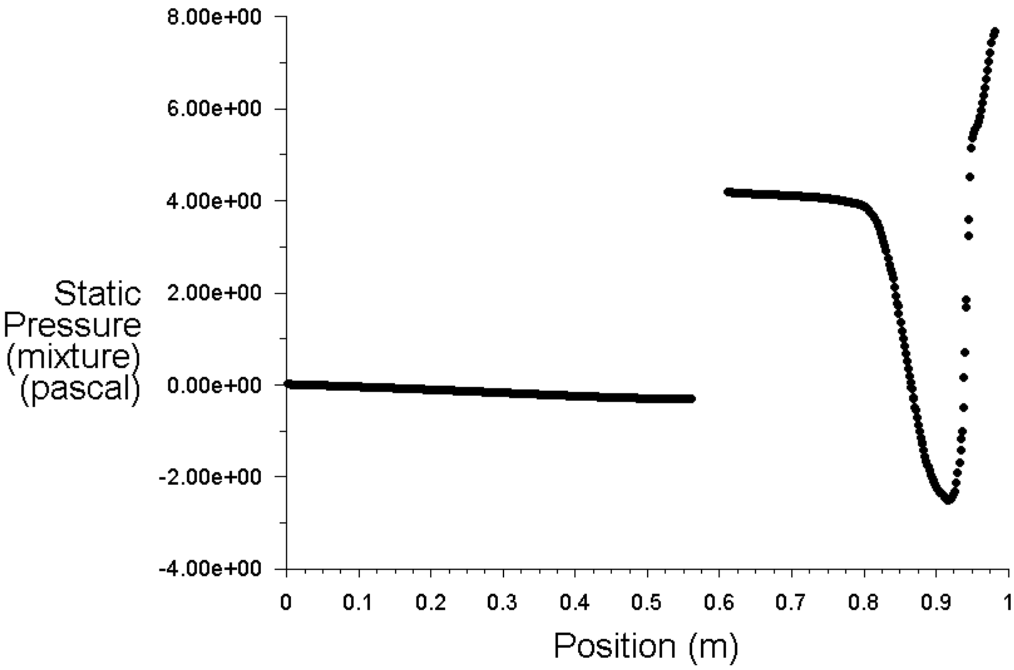


Figure 4.7: Variation of static pressure along the axis for the obstacle with diameter = 5 cm.

From the Figures 4.5, 4.6 and 4.7, it can be seen that as the size of the obstacle increases the pressure drop across it increases. It can also be seen from the Figures that the presence of obstacle affects the pressure distribution in the channel.

4.3.3 Effect of Diameter of the Obstacle on the Static Pressure Difference: In this section, the effect of size on static pressure difference has been explored. The static pressure difference has been calculated by the subtracting the area weighted average static pressure at the position of the obstacle from that at the outlet. It can be seen from Figure 4.8 that as the diameter increases the static pressure difference increases.

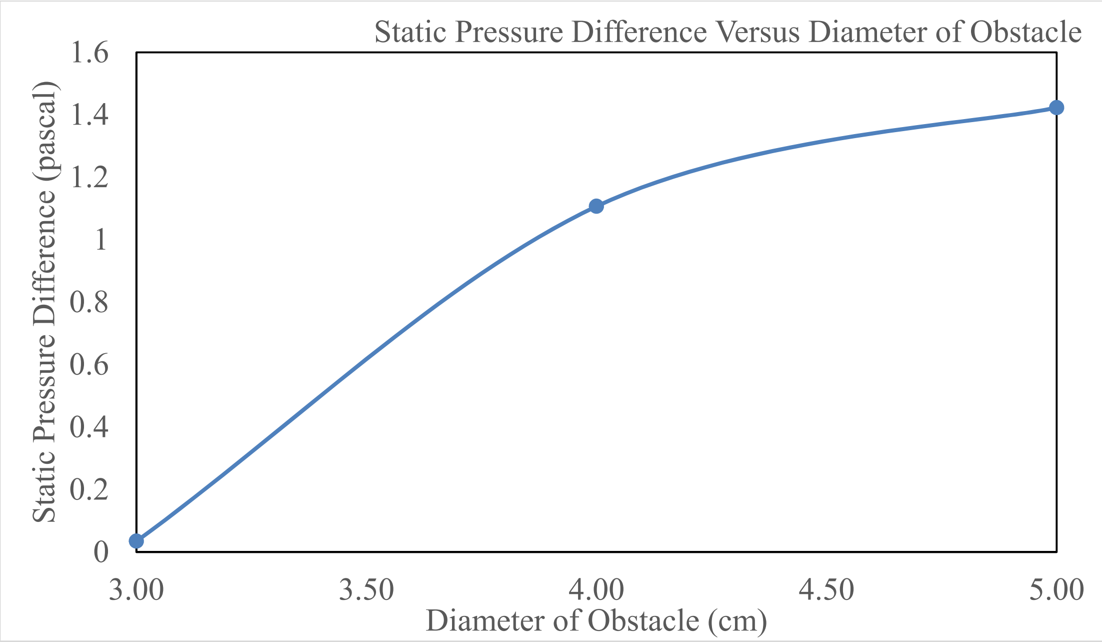


Figure 4.8: Variation of static pressure difference with diameter of the obstacle.

4.4 UNSTEADY STATE RESULTS

4.4.1 Effect of Diameter of the Obstacle on Flow Structure: In this section, the development of the flow structure with time has been shown in Figure 4.9 for the obstacle with diameter 3 cm. In the same way, Figure 4.10 and 4.11 shows the development of flow structure with time for the obstacle with diameter 4 cm and 5 cm respectively. For all the three cases considered here, air and water mixes in the enlarging section after entering from the respective inlet ports. The mixture velocity is decelerated in the enlarging inlet section. As the mixture approaches the obstacle, the phase distribution changes substantially from that in the enlarging section due to the changes in both magnitude and direction of the local velocities of the individual phases. This effect is becoming prominent as the size of the obstacle increases. It is clear from these Figures (Figures 4.9 to 4.11) that the flow becomes more unsteady and unstable as the size of the obstacle increases. It can also be noticed that vortices before and after the obstacle is becoming stronger with increase in size of the obstacle. Also, one more thing can be noticed. The phase distribution in the enlarging part is different for different cases.

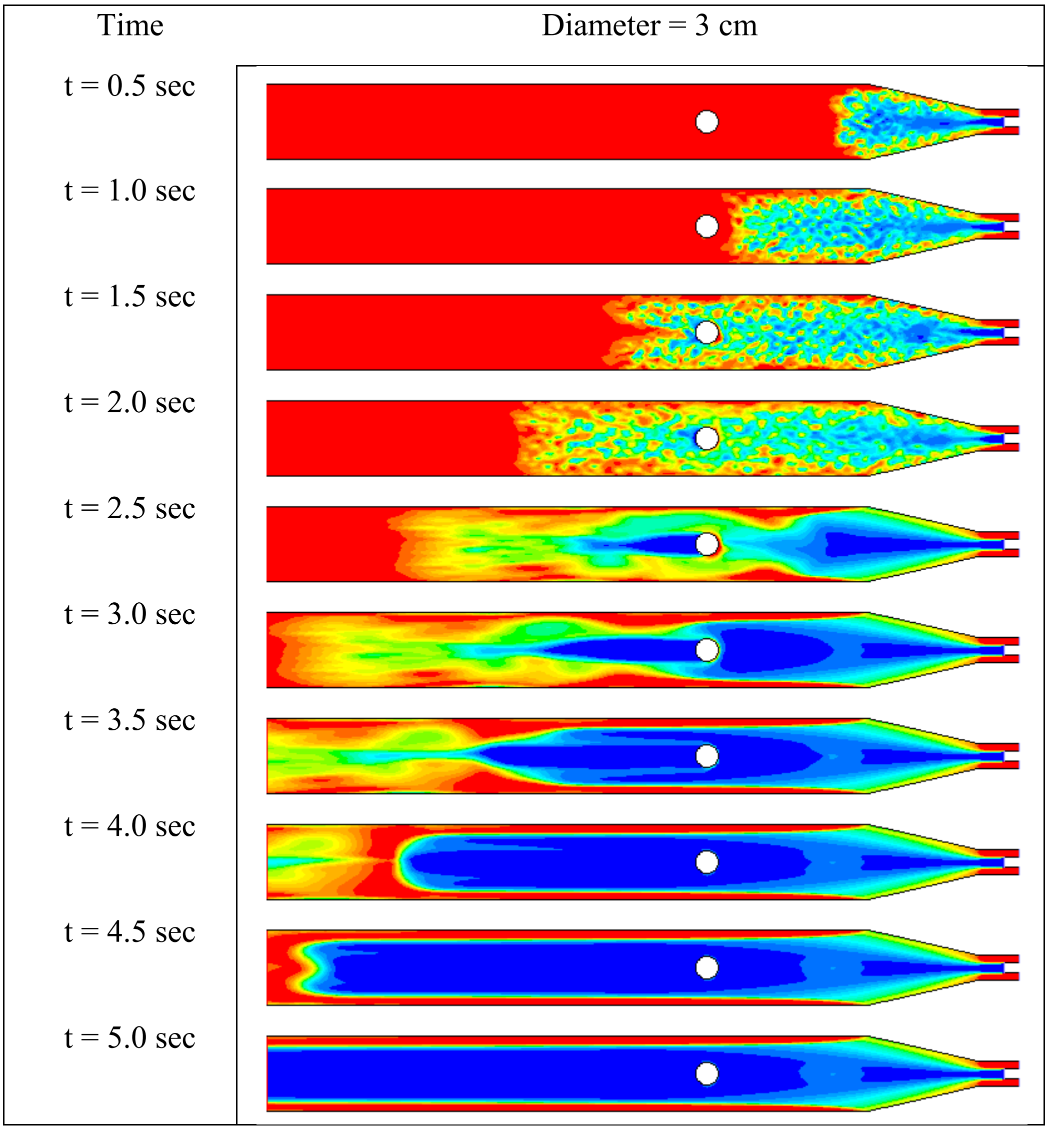


Figure 4.9: Flow structure at different time instants for the obstacle diameter = 3 cm

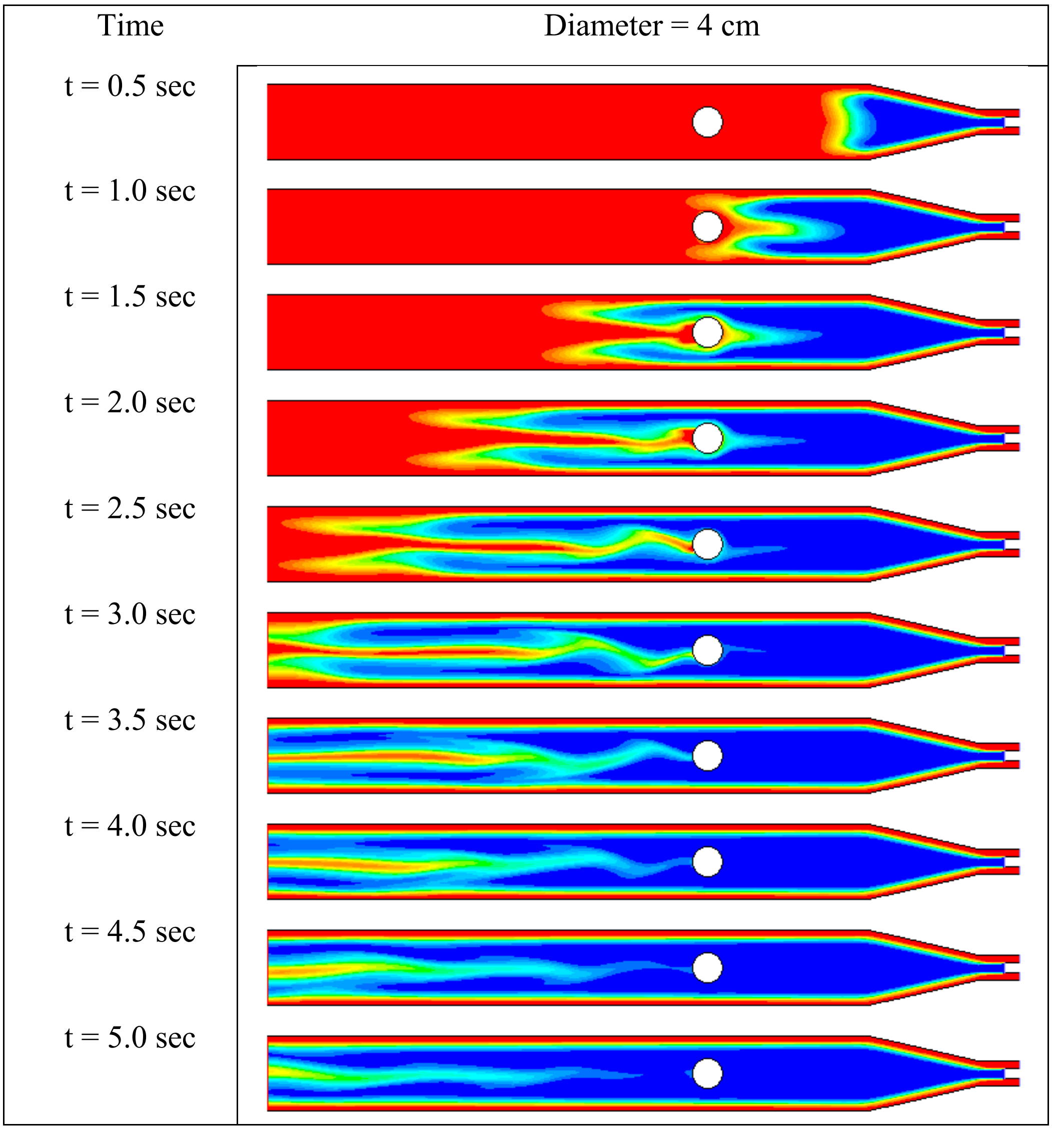


Figure 4.10: Flow structure at different time instants for the obstacle diameter = 4 cm

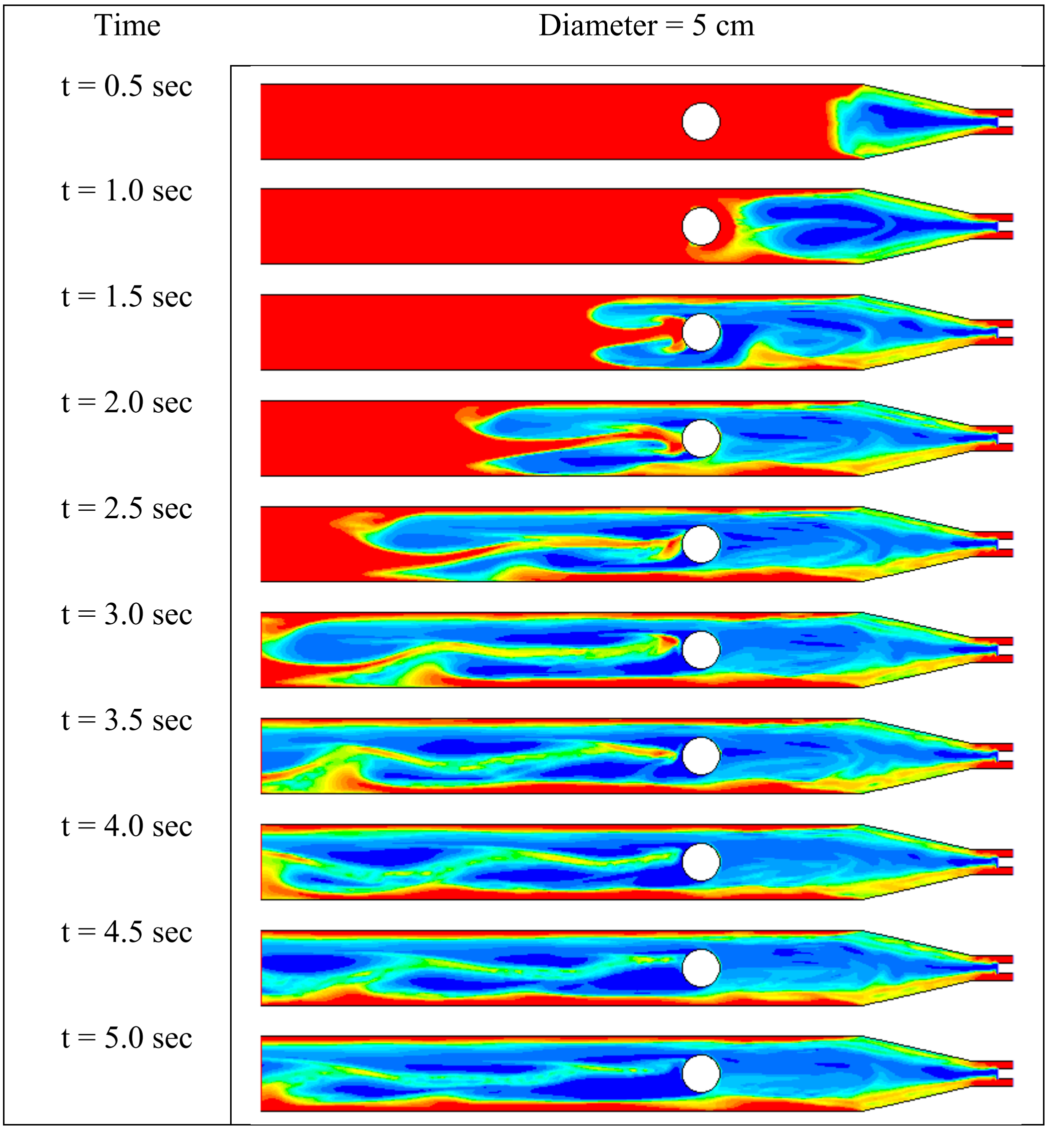


Figure 4.11: Flow structure at different time instants for the obstacle diameter = 5 cm.

4.4.2 Effect of Obstacle-Size on the Fluctuation in Static Pressure Gain: In this section, the gain in static pressure from the obstacle-position to the outlet have been calculated for each time step and plotted against time as shown in Figures 4.12, 4.13 and 4.14 for the obstacle size 3 cm, 4 cm and 5 cm respectively. It can be seen from these Figures that as time passes the gain in static pressure decreases.

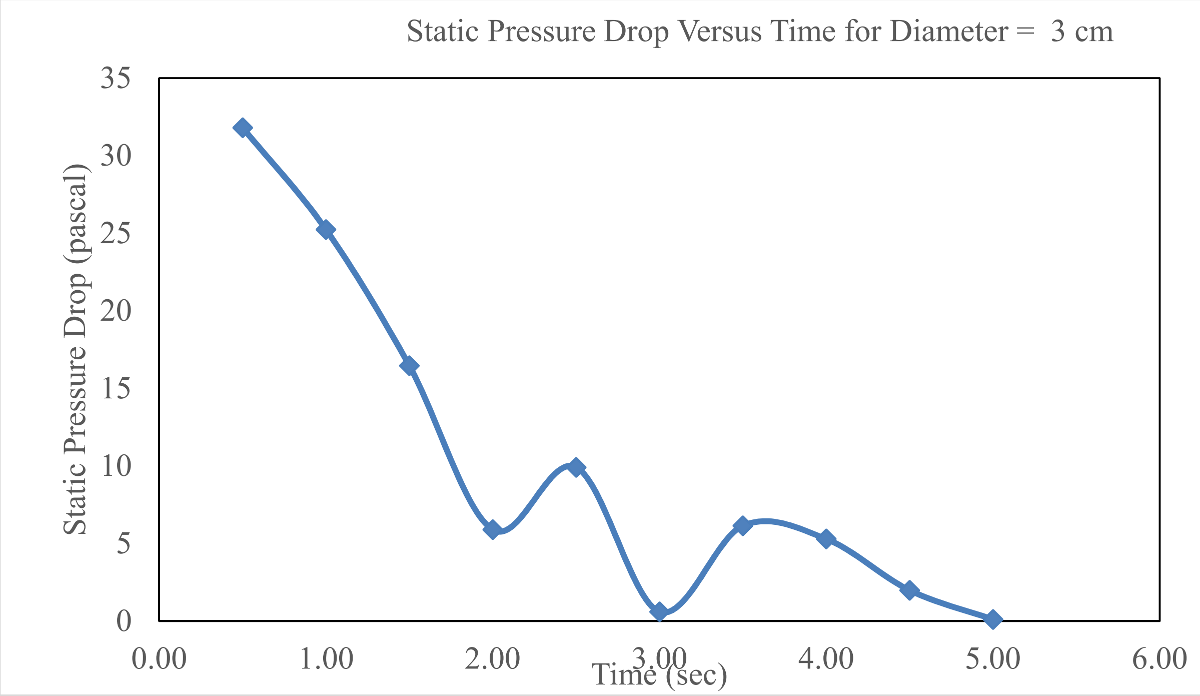


Figure 4.12: Variation of static pressure difference with time for diameter of obstacle = 3 cm.

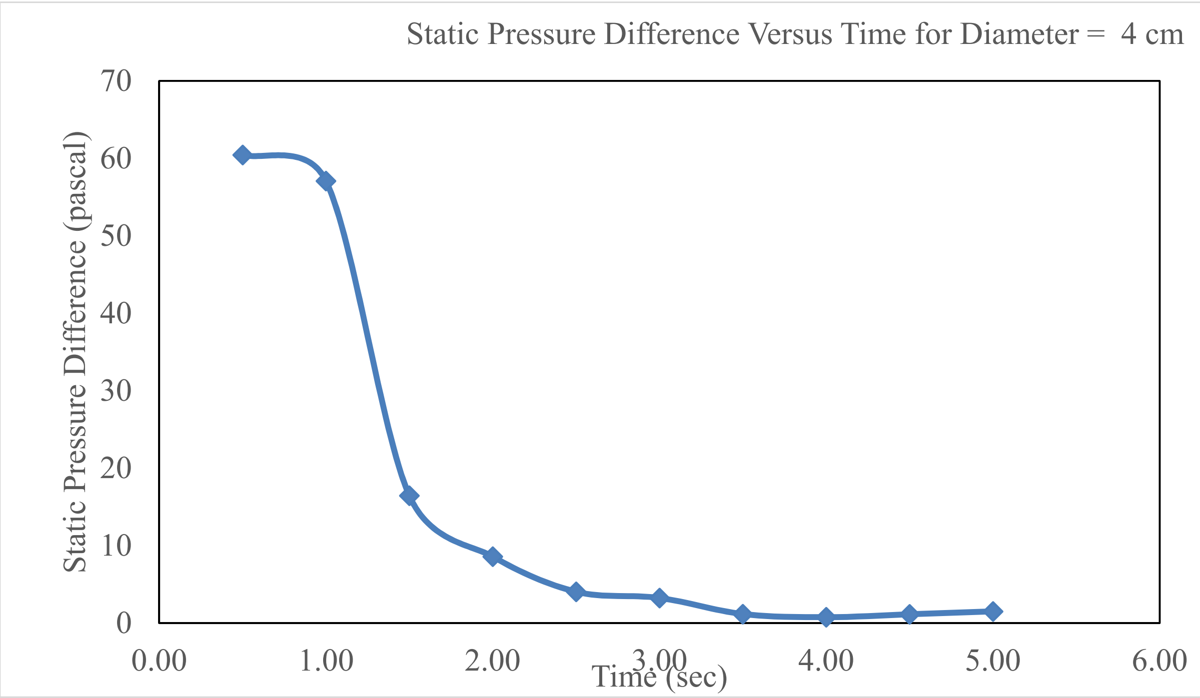


Figure 4.13: Variation of static pressure difference with time for diameter of obstacle = 4 cm.

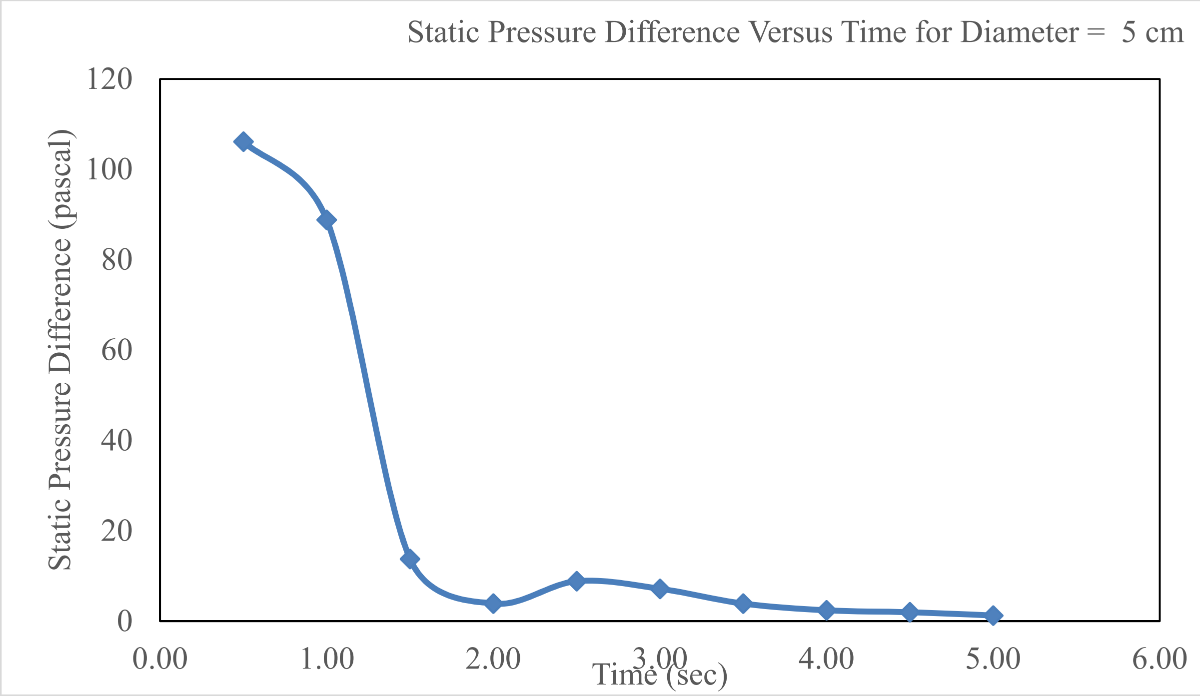


Figure 4.14: Variation of static pressure difference with time for diameter of obstacle = 5 cm.

CHAPTER 5

CONCLUSION

AND

SCOPE OF FUTURE WORK

5.1 CONCLUSION

An attempt has been made to study the effect of a cylindrical obstacle on the flow structure of air-water two-phase flow through a rectangular enlarging channel using numerical simulation. Basically, effect of the diameter of the obstacle on the flow structure, pressure fluctuation and pressure distribution was made. For the present numerical simulation, Finite volume method (FVM) with mixture model and Eulerian model have been used. Both steady and unsteady cases were considered during study. A marked difference in phase distribution around the obstacle was observed. Vortices before and after the obstacle became stronger with increase in size of the obstacle. It was found that as the obstacle size increases, the pressure drop increases and pressure distribution changes drastically. Some important findings obtained through the present study are given below.

- As the size of the obstacle increases, the phase distribution pattern around the obstacle also changes remarkably.
- Presence of the obstacle in the channel affects the pressure drop across the channel. As the size of the obstacle increases the pressure drop across it increases.
- Presence of the obstacle in the channel affects the pressure distribution across the channel.
- The fluctuation in pressure drop (against time) is more for obstacle with larger diameter.
- The existence of cylindrical obstacle generates vortices and enhances mixing.

5.2 SCOPE OF FUTURE WORK

In the present study, only the size of the obstacle was taken into consideration. There are many other factors affecting the flow regime and pressure drop (including pressure distribution) such as number of obstacles, shape of obstacle, fluid flow rate, etc, which can be considered in future. In future, one can take an attempt to capture the phenomenon considering 3D analysis.

REFERENCES

Bon, G.K., Numerical study of two phase air water interfacial flow: Plunging wave breaking and vortex interface interaction. Ph.d Thesis, University of Iowa.

Deniz, E., Eskin, N., 2011. Numerical analysis of adiabatic two-phase flow through enlarging channel. Istanbul Technical University, Istanbul, Turkiye.

Eames, I., Hunt, J.C.R., Belcher, S.E., 2004. Inviscid mean flow through and around groups of bodies. Journal of Fluid Mechanics 515, 371-389.

Esam, M.A., Riyadh, S.A., 2012. Experimental study of two phase flow around hydrofoil in open channel. Journal for Mechanical and Material Engineering 12, 72-86.

Habeeb, L.J., Riyadh, S.A., 2013. Experimental study and CFD simulation of two phase flow around multi-shape obstacles in enlarging channel. American Journal of Mechanical Engineering 8, 470-486.

Meng, H., On dispersed two phase flows past obstacles. Eindhoven University of Technology, 1993.

Talimi, V., Muzychka, Y.S., and Kocabiyik, S., 2012. A review on numerical studies of slug flow hydrodynamics and heat transfer in micro-tubes and micro-channels. International Journal of Multiphase Flow 39, 88-104.

Zhang, X.B., Qiu, L.M., Gao, Y., Zhanga, X.J., 2008. Computational fluid dynamics study on cavitation in liquid nitrogen. Cryogenics 48, 432-438.

Zhou, H., Mo, G., Cen, K., 2011. Numerical investigation of dispersed gas–solid two-phase flow around a circular cylinder using Lattice Boltzmann Method. Computers & Fluids 52, 130-138.